Glenn Research Center

WIND Tutorial Case

Glenn Research Center

Solution Process

This tutorial is documented in full in the *WIND User's Guide*:

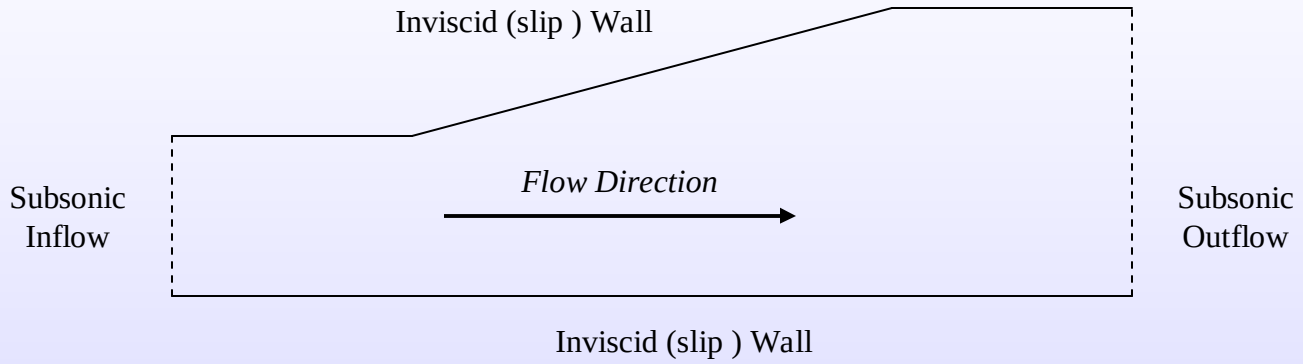
www.grc.nasa.gov/www/winddocs/user/tutorial.html

Solution Process:

1. Gather Information
2. Create the Computational Grid
3. Set the Boundary Conditions
4. Set the Initial Conditions
5. Set the Program Control Parameters
6. Run WIND
7. Monitor the Convergence
8. Examine the Results

Glenn Research Center

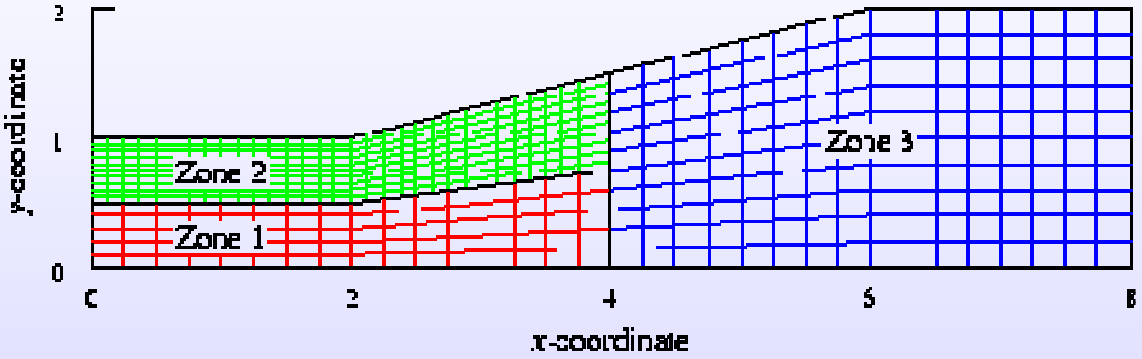
Gather Information



- Subsonic internal flow in a diverging duct.
- Want to know the static pressure distribution and mass flow rate within 10%.
- Inlet flow conditions: $M = 0.78$, $p_t = 15$ psi, $T_t = 600$ ^{o}R.
- Reynolds number $Re = 3.023 \times 10^5$.
- Exit static pressure $p = 14.13$ psi.

Glenn Research Center

Create the Computational Grid

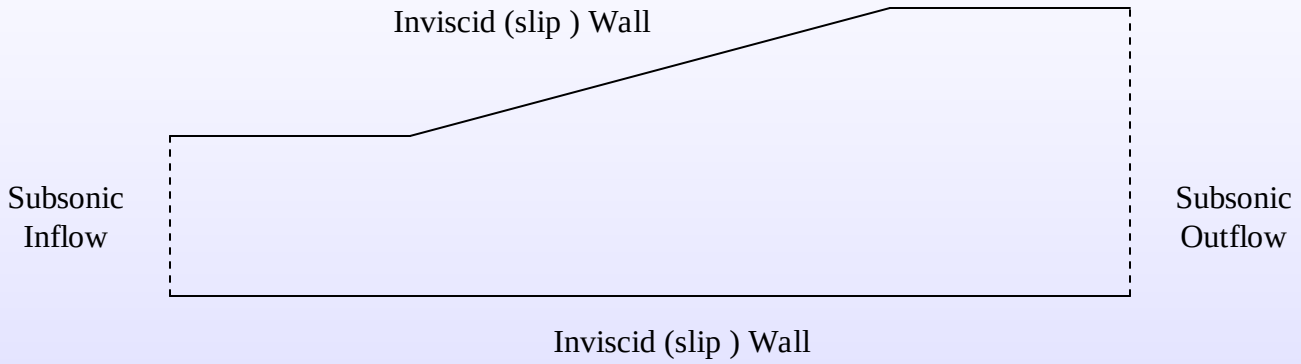


Test case grid

- 3 zones: 17 × 6, 33 × 11, 17 × 11.
- Grid created algebraically and written in PLOT3D xyz format.
- Converted to a Common Grid (case4.cgd) file using CFCNVT.

Glenn Research Center

Set the Boundary Conditions



- GMAN is used in graphical mode to set boundary condition types:
  - Subsonic Inflow: Arbitrary Inflow BC
  - Subsonic Outflow: Outflow BC
  - Walls: Inviscid Walls BC
- Zone interfaces identified automatically and coupling information was written to the Common Grid (case4.cgd) file.
- Further boundary condition inputs were specified in the Input Data file (case4.dat).

Glenn Research Center

Set the Initial Conditions

Create the initial flow field as a uniform flow using the freestream conditions as set by the freestream keyword in the input data file (case4.dat).

Glenn Research Center

Set the Program Control Parameters

Physical Model Controls:

- Dimensionality (3D, 2D, axisymmetric)
- Flow Equations (Euler, PNS, thin-layer NS, RANS)
- Turbulence Model
- Gas Model & Chemistry

Numerical Model Controls

- Time Stepping
- Explicit and Implicit Numerical Operators
- Damping Scheme
- Convergence Acceleration
- Convergence Monitoring Parameters

Initial flow field set by freestream: Mach, p (psi), T (deg R), alpha, beta

Exit static pressure (psi)

```
WIND test case #4, 2-D, 3 zones
Subsonic internal flow
Run 1

/ Inlet conditions
Freestream total 0.78  15.  600.  0.  0.

/ Boundary conditions
Downstream pressure 14.13 zone 3

/ Numerics
Cycles 1000

/ Viscous terms
Turbulence euler

/ Convergence data
Loads
    print planes frequency 5
    zone 1
        surface i 1 mass
    zone 2
        surface i 1 mass
    zone 3
        surface i 1     mass
        surface i last mass
Endloads

End
```

Run 1000 cycles

Solve inviscid equations

Monitor convergence by computing the mass flow at entrance and exit planes.

Glenn Research Center

Run WIND

- Run Wind-US using the WIND script.

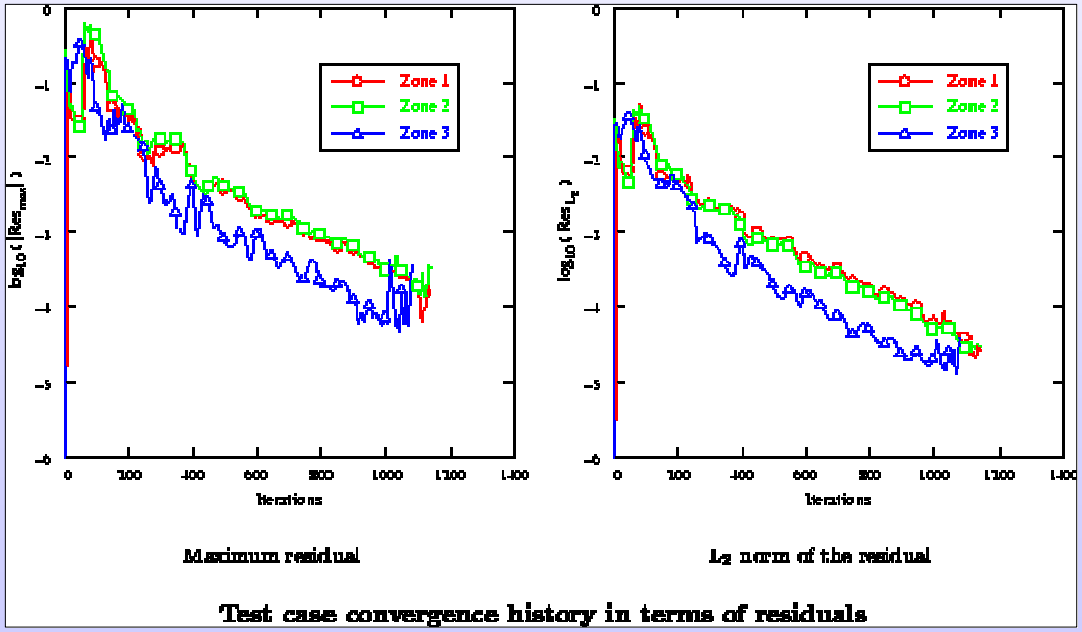
  wind –runinplace
- Specify use of Wind-US 2.0
- Specify name of input data file prefix, case4.
- Run interactively.
- Default names for solution, grid, and output files.
- Initial flow solution created.
- List Output to a file, case4.lis.

```
> wind -runinplace
Running command line version of WIND.

                  ***** WIND Run Script *****

Current wind settings are:

--Wind test mode set to NODEBUG
--Wind debugger set to DEFAULT
--Wind run que set to PROMPT
--Wind run in place mode is set to YES
--Wind multi-processor mode set to NO
--Wind run directory set to PROMPT
--Wind bin directory set to /net/zargon/usr2/wind/wind

                Select the desired version

  0: Exit wind
  1: alpha version
  2: Version 2.0
  3: Version 3.0
  4: Version 4.0
  5: Version 5.0

Enter number or name of executable.......[5]: 5
Basic input data................(*.dat): case4
Output data.............(*.lis,<CR>=case4):
Mesh file...............(*.cgd,<CR>=case4):
Flow data file..........(*.cfl,<CR>=case4):

************************************************************
case4.cfl does not exist, a fresh start will be performed.
************************************************************

Enter a queue number from the following list or <CR> for default:
  1: REAL (interactive)
  2: AT_QUE

Queue name.....................(<CR> for 1): 1

Print output at screen?.................(y/n,<cr>=y): n

Version..............: wind5.exe
Input file name.......: case4.dat
Wind output to........: case4.lis
Grid file name........: case4.cgd
Flow file name........: case4.cfl
Job run que type is...: REAL
```

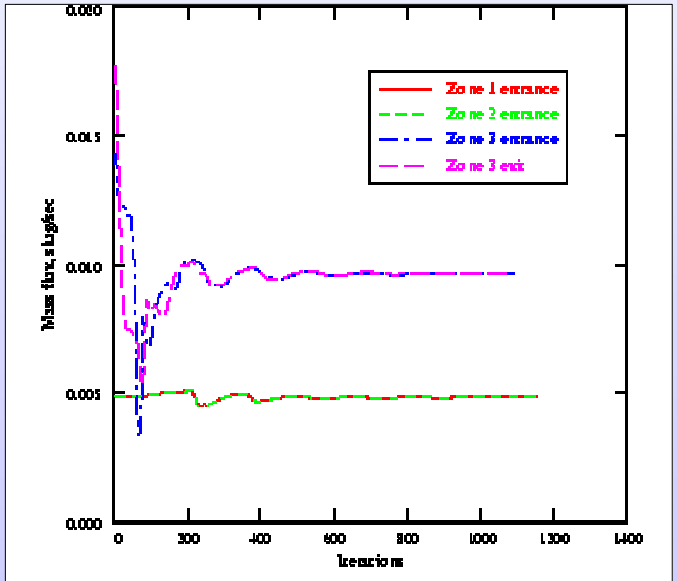
Glenn Research Center

Monitor the Convergence

- RESPLT used to extract residuals and mass flow from List Output file (case4.lis) and to create a GENPLOT file.
- CFPOST can be used to plot the GENPLOT files containing convergence data.



Test case convergence history in terms of residuals

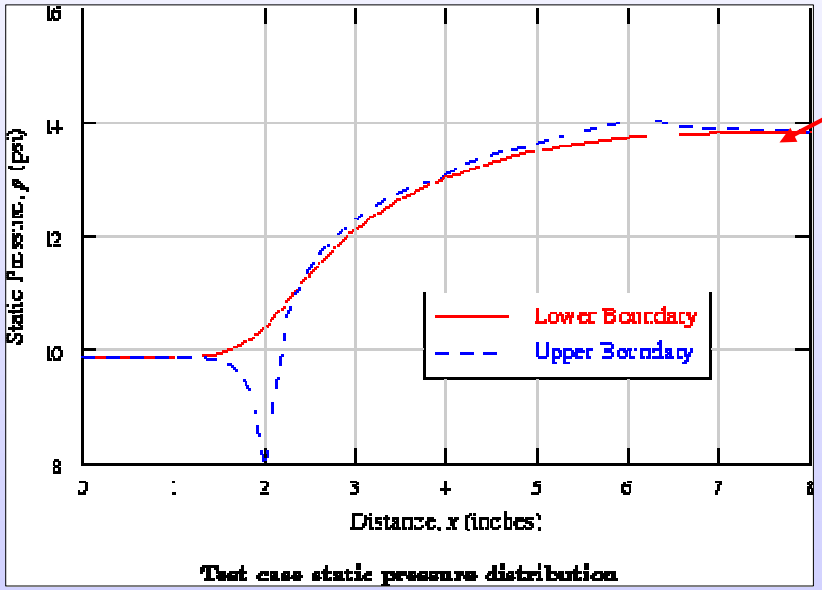


Test case convergence history in terms of mass flux

Glenn Research Center

Examine the Results

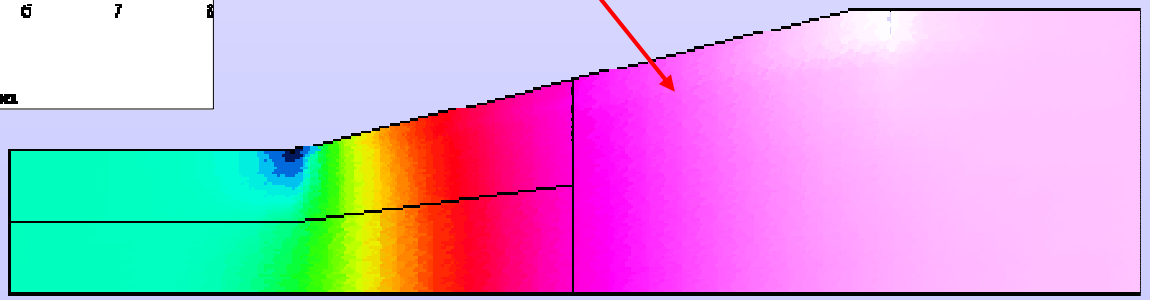
- Computed results stored in the Common Flow (case4.cfl) file.
- Results processed using CFPOST.



Test case static pressure distribution

Wall static pressure distribution

Static pressure field



Test case constant static pressure contours